THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE V MACHINE BUILDING TECHNOLOGY. ISSN 1221-4566
ANUL XIX (XXIV) - 2001

A SIMULATION MODEL FOR COLD PLASTIC DEFORMATION OF HEATER EXCHANGER PLATES

Assoc. Prof. Dr. Eng. Viorel Păunoiu
Prof. Dr. Eng. Dumitru Nicoară
Prof. Dr. Eng. Mircea Modiga
"Dunărea de Jos" University of Galaţi

ABSTRACT

The heater exchanger plates are products obtained by cold plastic deformation of sheets using complex tools. In their service, due to environmental conditions, due to residual stresses induced in material as a result of deformation and material itself, it is possible to appear their crack and fracture. In the paper it is presented, based on a simulation model, the effect of residual stresses toward this phenomenon.

1. Introduction

The heater exchanger plates are complex parts both from design and function point if view (figure 1).

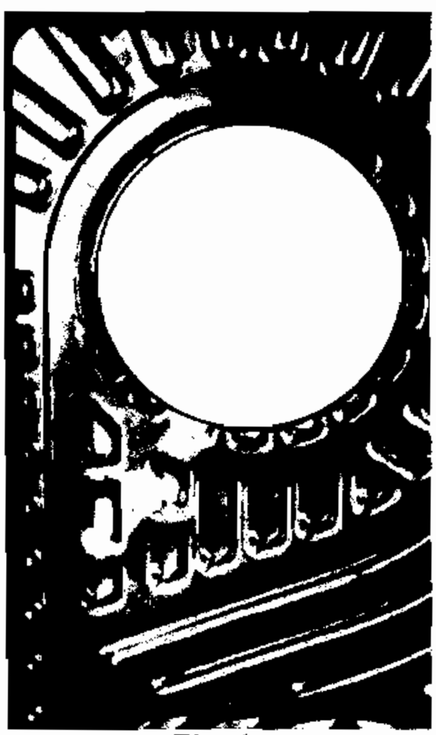

Fig. 1

Obtained by cold plastic deformation, they are assembled in a package, which has the function to exchange the heat between a warm and a cold agent.

In service, it was observed the plates crack in the region of worm agent entrance.

The paper presents an analysis of the stress and strain state as it results after the deformation, to establish its influence in crack appearance.

2. Elements of Heater Exchanger Plates Manufacturing

The plates are obtained by plastic deformation in a drawing operation. The active elements are complex parts, because the geometric form of the plates is also, complex. The entire part surface is deformed. The degree of deformation is small, even in the region of crack appearance.

The material used for obtaining the plates is stainless steel, with a bigger tendency of straightening by plastic deformation.

3. Simulation Program Background

CAE applications for die design and sheet metal forming process simulation provide a tool to assist die designers and stamping engineers to evaluate the manufacturing feasibility of parts at the design stage; to explore alternative designs and evaluate trade-offs, and eventually, to derive an optimised design using the right material processed in a cost effective and timely fashion.

Finite Element Analysis is a powerful simulation tool for analysing complex three-dimensional sheet metal forming problems.

The program used in present simulation has two parts: a pre-processing part and a post-processing part.

The post-processing package has the ability to process the CAD surface using wire frame data, generate suitable meshes for sheet metal forming applications and modify CAE models for design iteration studies.

The post-processing package has the ability to process the analysis results and to produce a meaningful conclusion.

In the present case, it was elaborated a model which comprise an extended region from the part which include the crack zone (figure 2). The blank is a circular sheet with hole.

Fig. 2

3. The Simulation Model Description

The model has as the basic geometric elements a number of 24 beads situated around a circular zone, similar with the region where the crack appears.

After the geometric profiles were building, they were meshed. The model has at the beginning of deformation 2500 elements.

In figure 3 is presented the punch model. The die model is similar with the punch model, with the accordingly clearance.

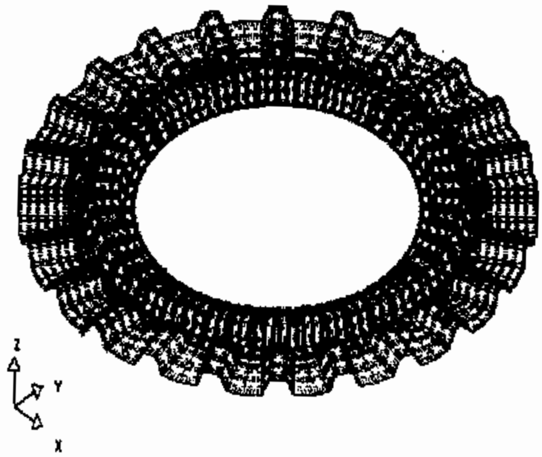

Fig. 3

In figure 4 is presented the blank model.

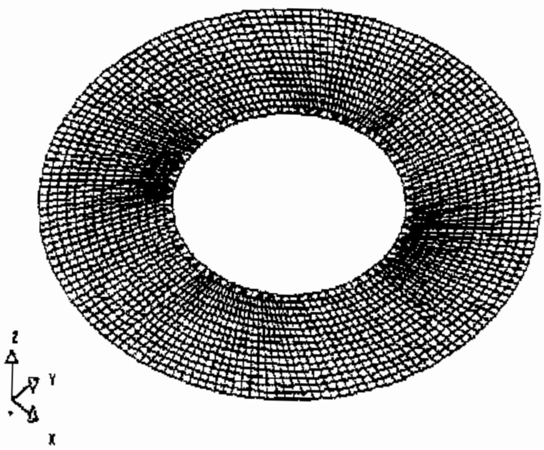

Fig. 4

5. The Simulation Parameters

The parameters used for simulation were:

- the friction coefficients between the blank and the active elements were equal with 0.11;
- the depth of deformation was 5.5 mm;

- the material has a stress/strain curve as it is presented in figure 5.

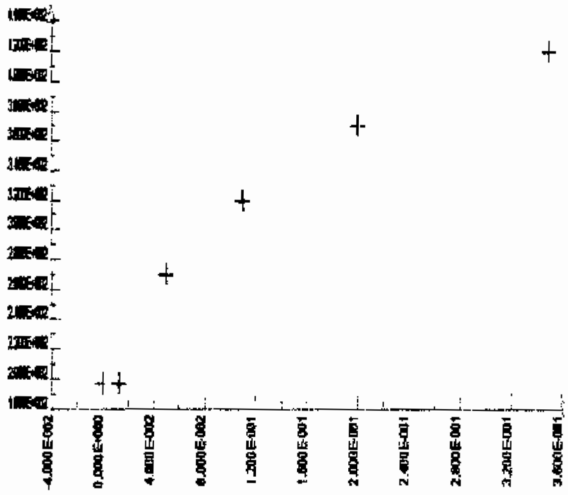

Fig. 5

6. The Results Analysis

As it follows, it will be presented the simulation results.

6.1. The Thickness Variation

In figure 6 is presented the thickness variation of the deformed part.

The thickness is variably, but the differences are very small. In the crack zone the thickness is almost the same with the blank thickness.

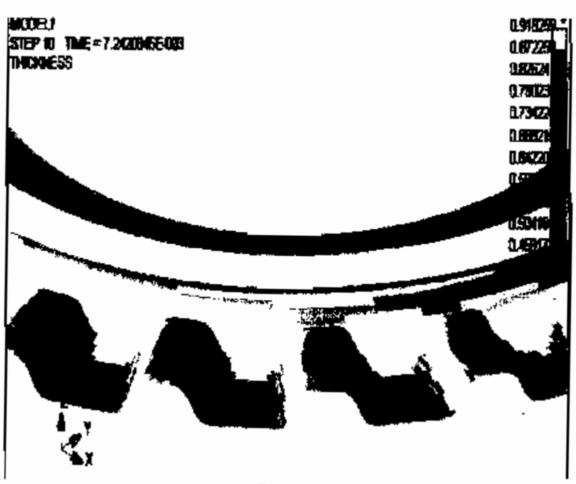

Fig. 6

6.2. The Radial Stress Variation

In figure 7 is presented the radial stress variation of the deformed part.

The radial stress variations show that:

In the crack zone, in the middle of the material thickness, the stresses are of tensile, of about 300 MPa. At the bottom, stresses are the same. In the front the stresses are of compression, which show a bending effect. The circumference stresses are of compression, on all the thickness. It results that an element is in pure shear, with the value of tangential stress near to the absolute values of radial and circumference stresses.

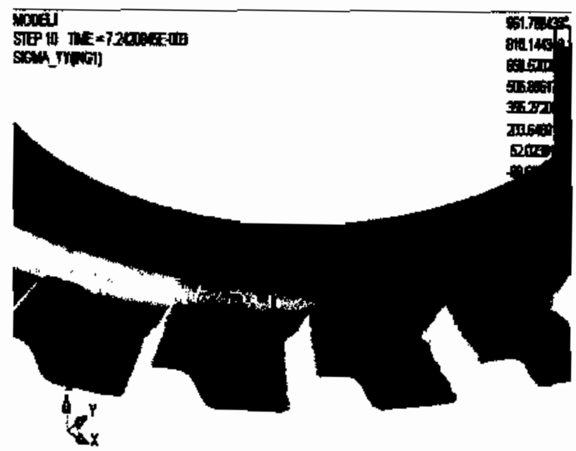

Fig. 7

This aspect is important because could be a factor for the crack appearance with a zigzag form, as it was observed to the visual examination.

6.3. Von-Mises Stress Variation

Von-Mises stress variation is presented in figure 8.

In accordance with Von-Mises criterion, the plastic state is reach when the flow stress is equal with:

$$\sqrt{2}\,\sigma_c = \sqrt{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2} \quad (1)$$

where: σ_1, σ_2, σ_3 are the normal, principal stresses.

The figure shows that all material is plastically deformed. The stresses state is different in different regions of the blank.

The maximum stresses never exceed the strength material resistance.

7. Conclusions

The present analysis shows that in the crack region the values of stresses not exceed the strength material resistance. As a result these stresses aren't the singular cause of the crack appearance. The material itself, the environmental condition could be also factors for crack initiation.

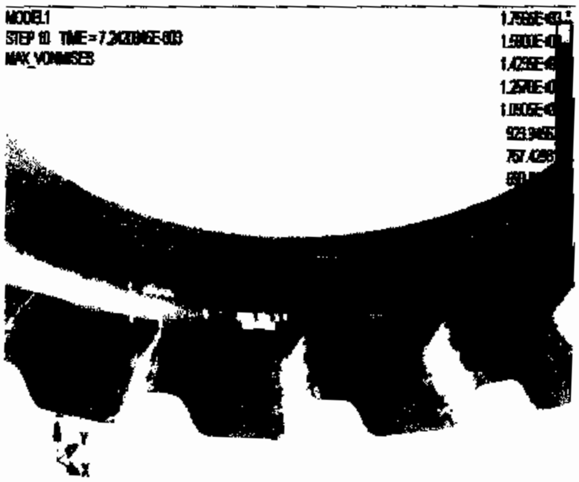

Fig. 8

REFERENCES

[1] Bathe, K., J. – Finite element Procedures in Engineering Analysis, Prentice Hall, 1982;

[2] Bellet, M., i.e.– Séminaire de plasticité et de mise en forme des métaux, CEMEF, 1994;

[3] Teodorescu, M., Zgură, Gh. – Tehnologia presării la rece, Editura Didactică şi pedagogică, 1980;

[4] Teodorescu, M., ş.a. – Prelucrări prin deformare plastică la rece, vol. I, Editura Tehnică, 1987

[5] Voinea, R., ş.a. – Introducere în mecanica solidului cu aplicaţii în inginerie, Editura Academiei, 1989

[6] Wagoner, R.H., Chenot, J.L. – Fundamentals of metal forming, John Wiley, New York, 1996.

[7] LS-DYNA Application Manual

UN MODEL DE SIMULARE A DEFORMĂRII PLASTICE A PLĂCILOR SCHIMBĂTOARELOR DE CĂLDURĂ

Rezumat

Plăcile schimbătoarelor de căldură sunt repere obţinute prin deformare plastică, utilizând echipamente complexe. În timpul lucrului, datorită factorilor de mediu, a tensiunilor induse în urma deformării plastice şi a materialului însuşi, este posibil să apară fisurarea sau ruperea acestora. În lucrare se prezintă, pe baza unui model de simulare în element finit, efectul tensiunilor reziduale în apariţia acestui fenomen.

UN MODÈLE DE SIMULATION POUR LA DÉFORMATION EN PLASTIQUE À FROID DES PLATS D'CÉCHANGEUR DE RÉCHAUFFEUR

Résumé

Les plats d'échangeur de réchauffeur sont des produits obtenus par la déformation en plastique à froid des feuilles à l'aide des outils complexes. Dans leur service, dû aux conditions environnementales, dues aux efforts résiduels induits en matériel en raison de la déformation et matériel lui-même, il est possible d'apparaître leur fente et rupture. Dans le papier il est présenté, basé sur un modèle de simulation, l'effet des efforts de résiduel vers ce phénomène.